Course Material
Onshape

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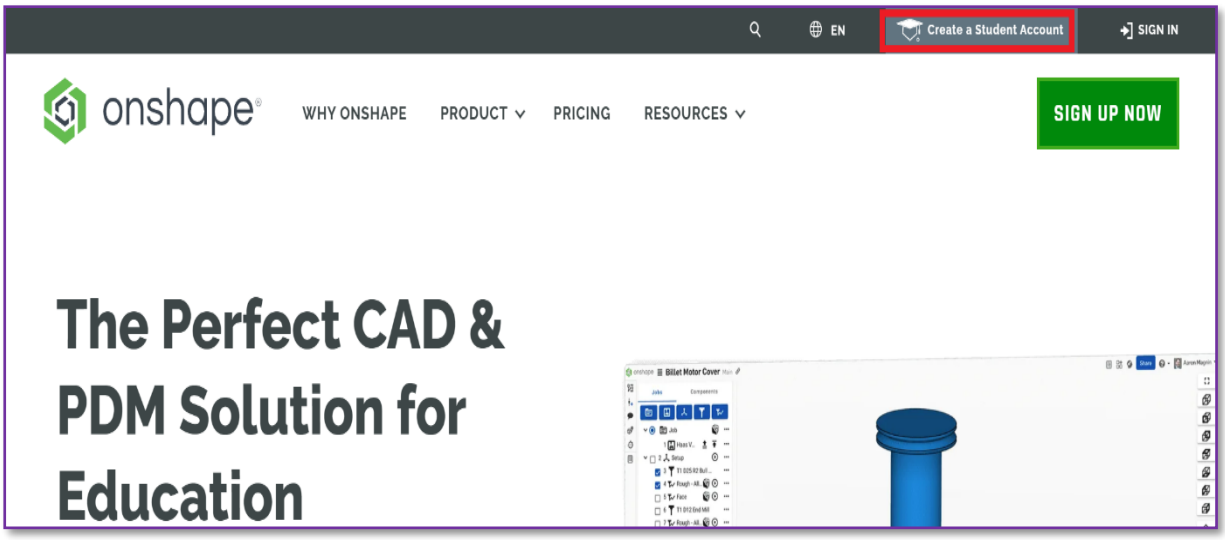
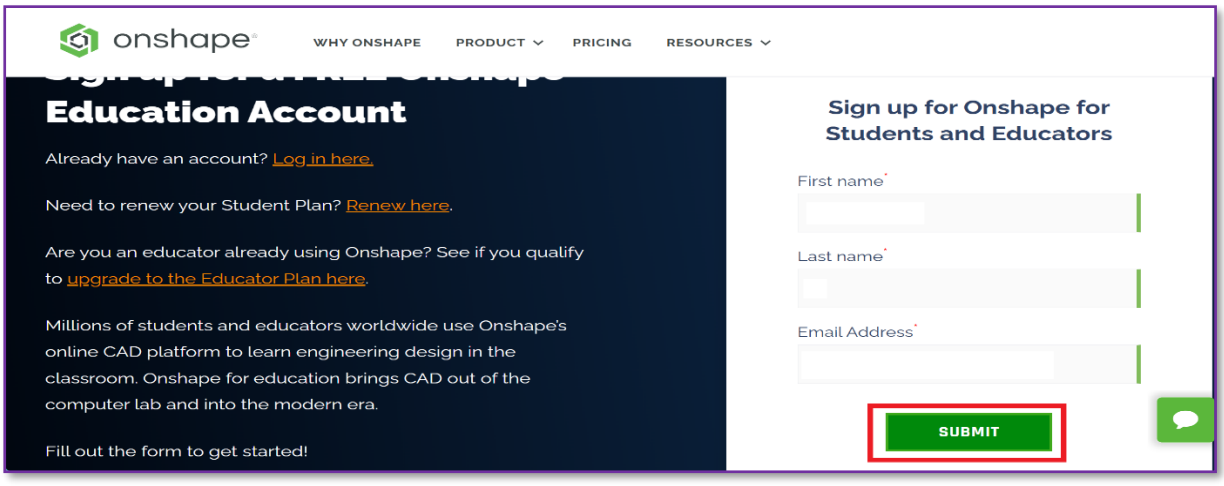
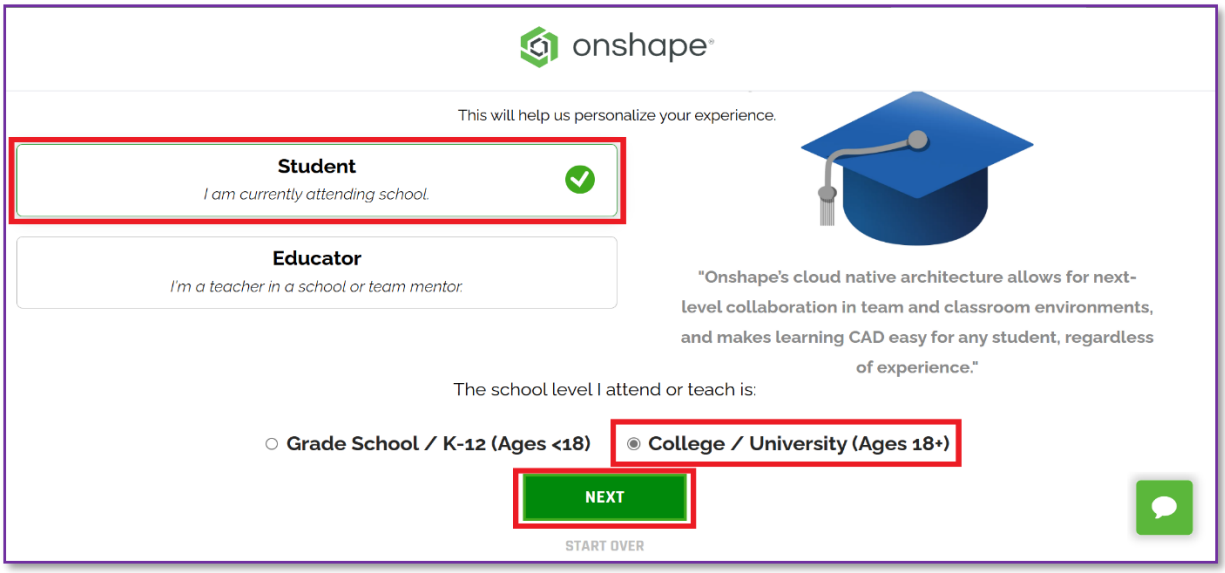
Installation

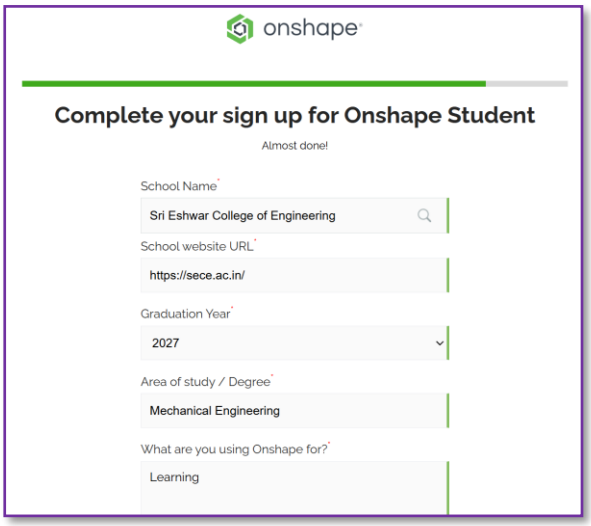
1. Go to https://www.onshape.com/en/
2. Click on **"Create a Student Account"** → Select **"Windows Installer (.exe)"**.

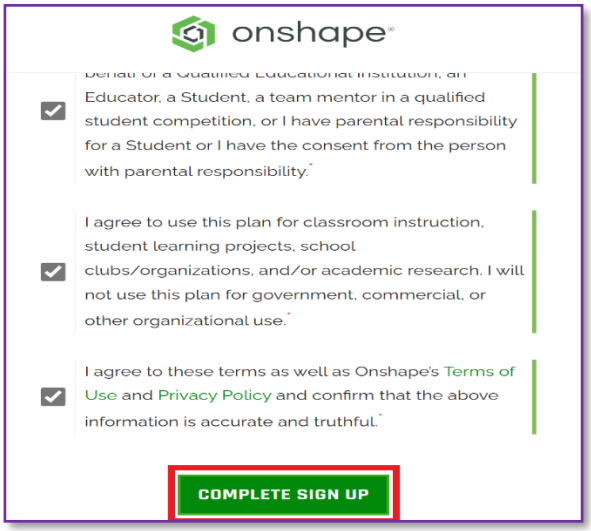


3. Enter your student credentials and complete the sign-up process.

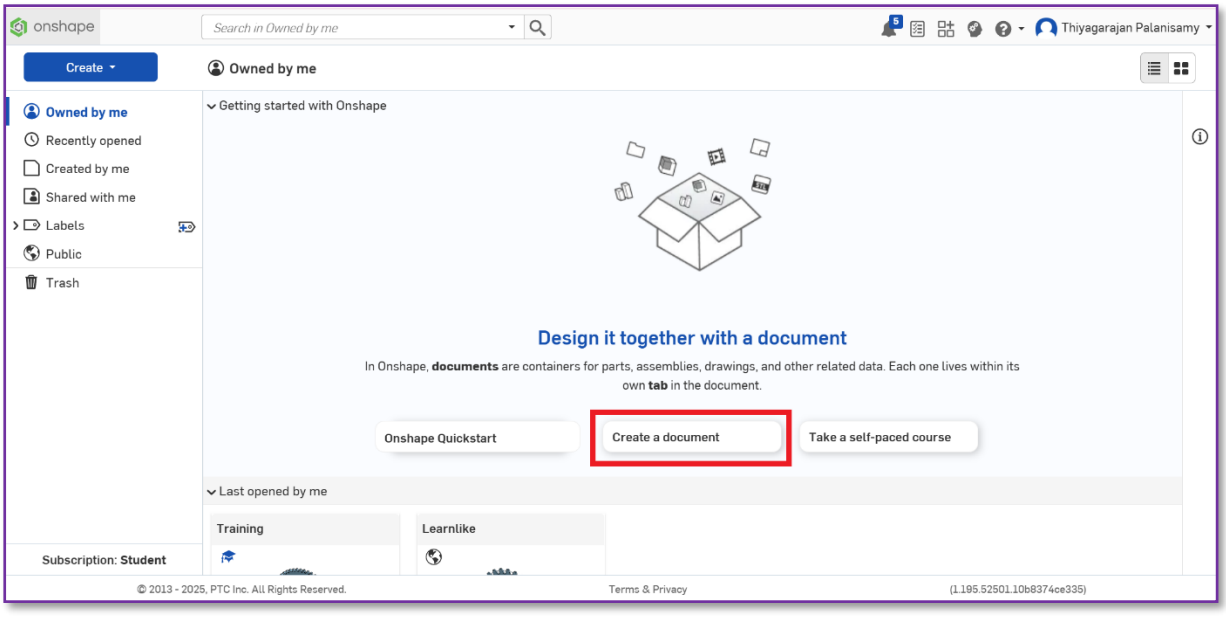




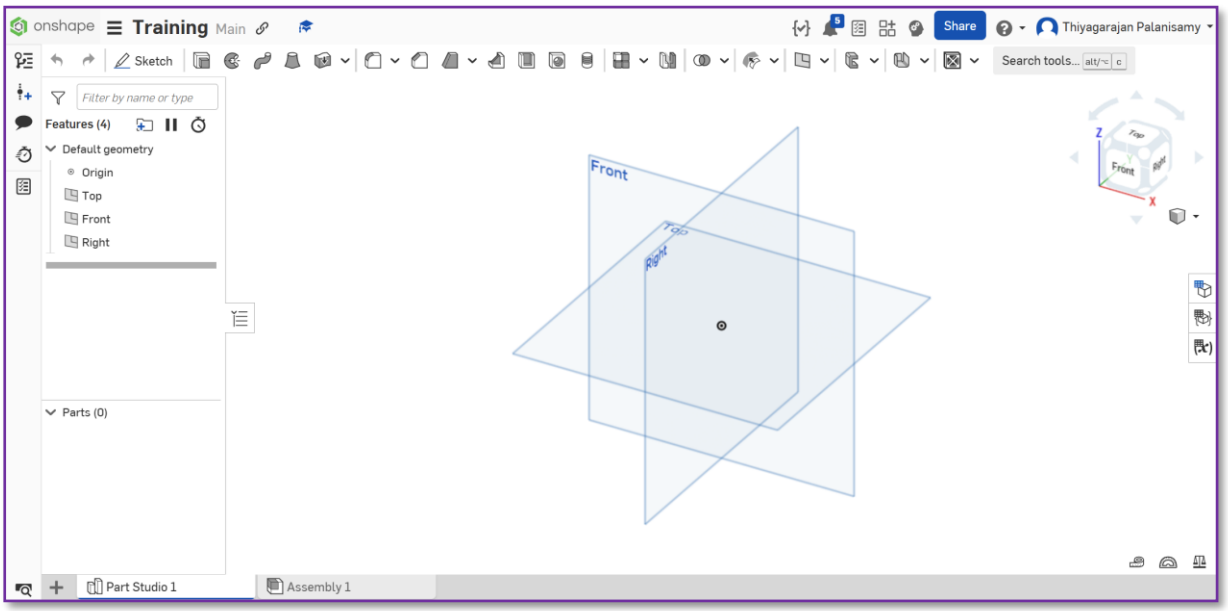




4. After sign-up, you will be redirected to the Onshape dashboard. Click on "**Create a document**".



5. Onshape web application will launch with the given project name.

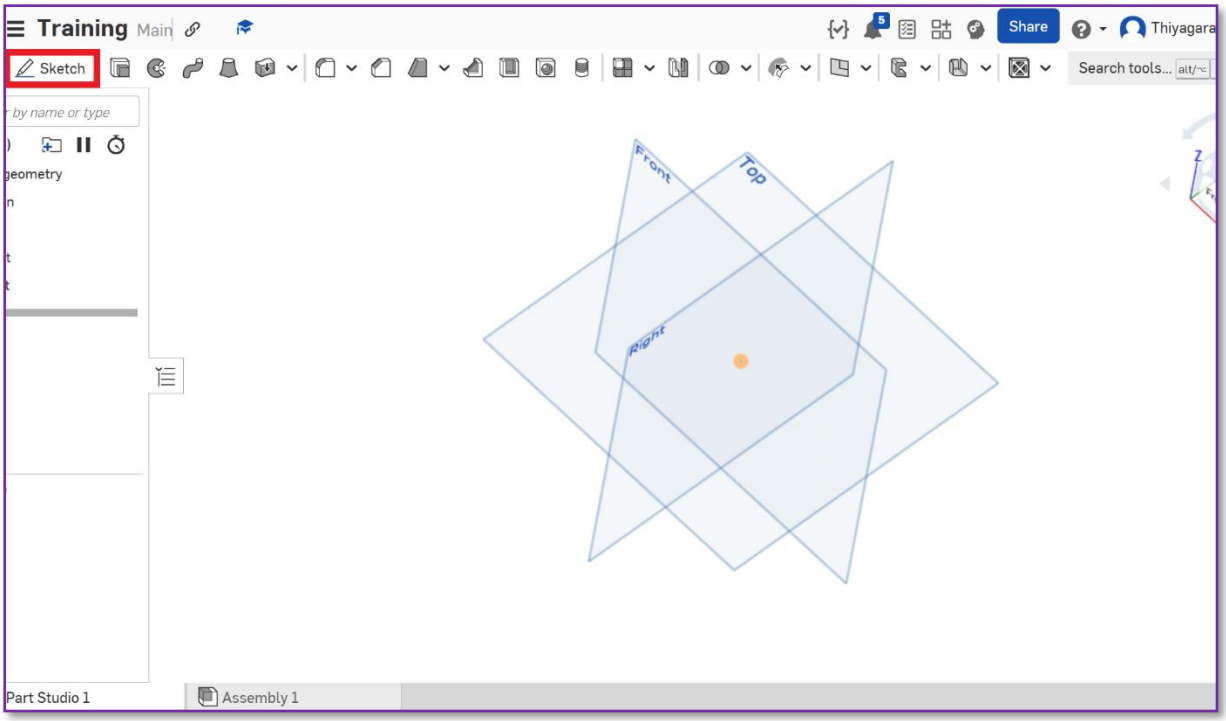


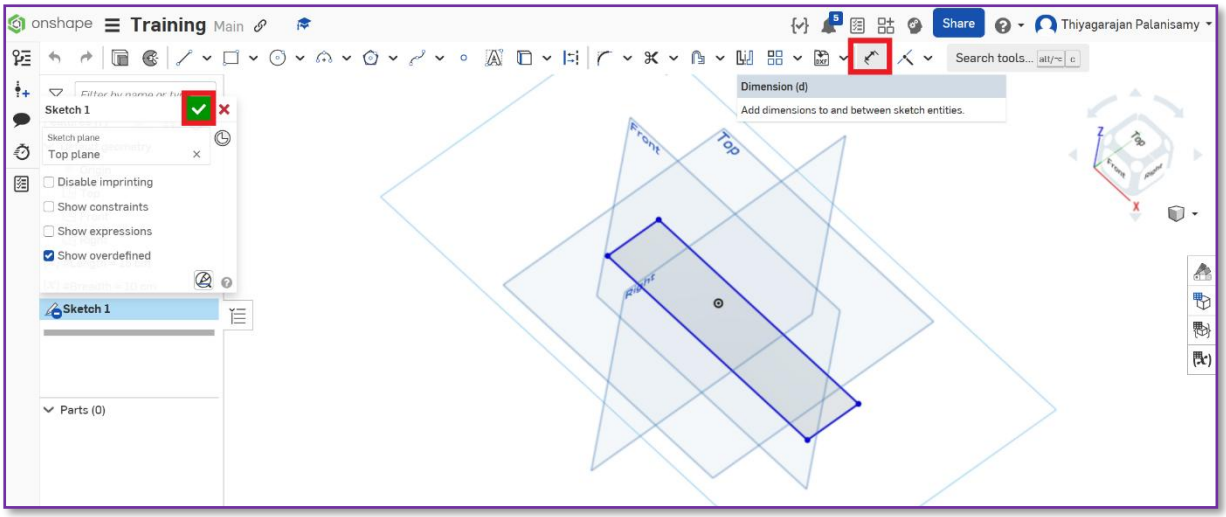
Basic Navigation Controls

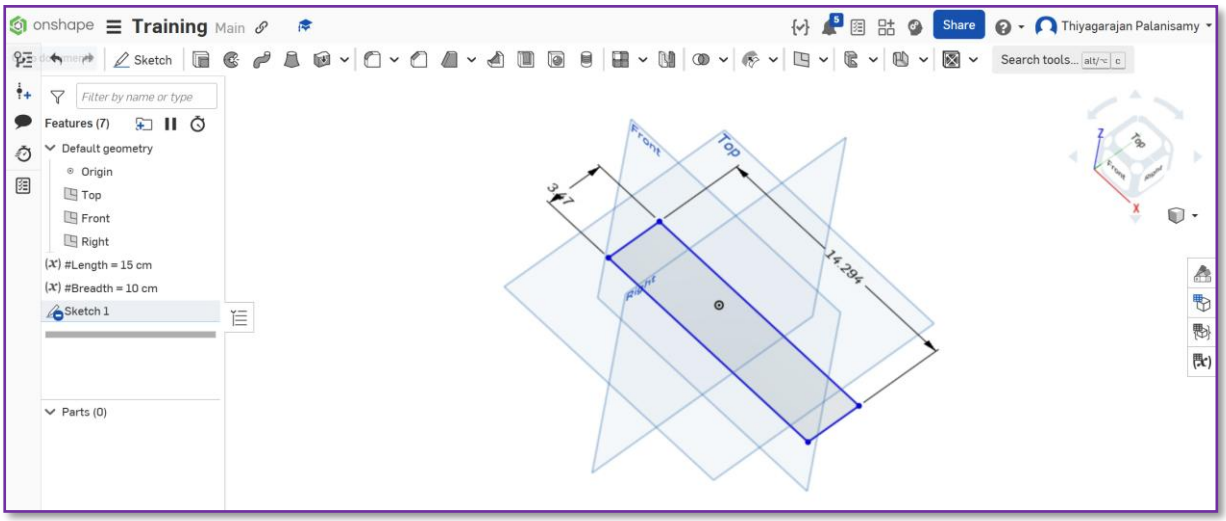
- **Zoom** → Use the **mouse scroll** to zoom in and out.
- **Free Rotate** → Hold **right mouse button** to freely rotate the view.
- **Pan** → Hold **middle mouse click** to pan across the workspace.

Onshape – Variable Table

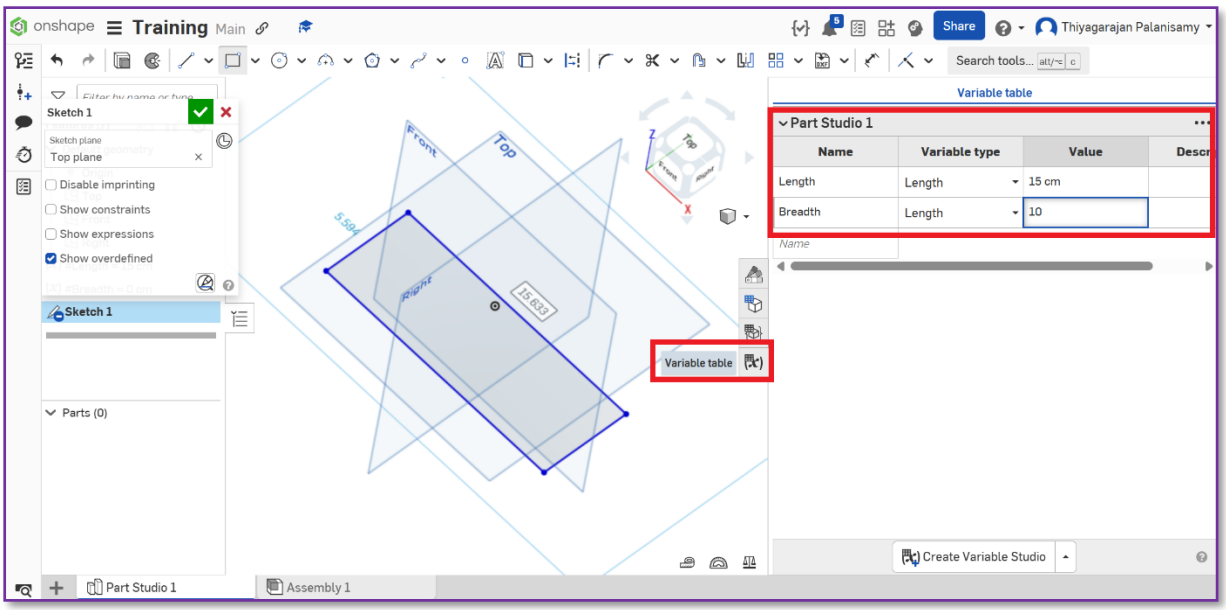
1. Click on "**Sketch**" and choose a plane to draw any shape. Add dimensions to the sketch and click on the **"tick mark"** to complete the sketch.



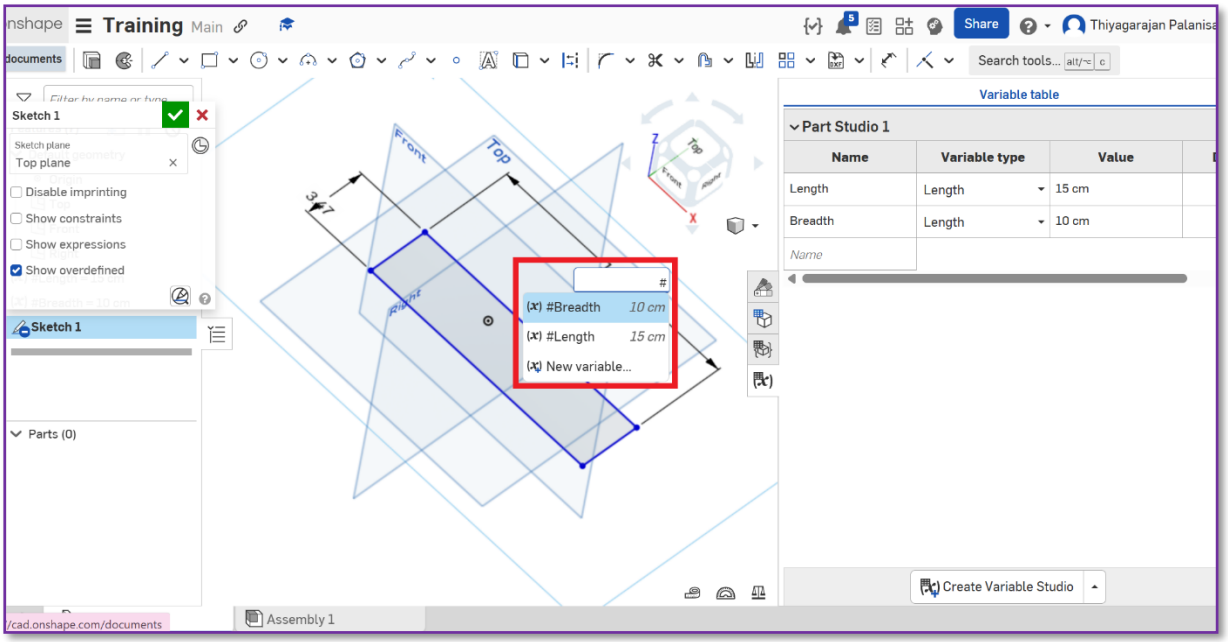




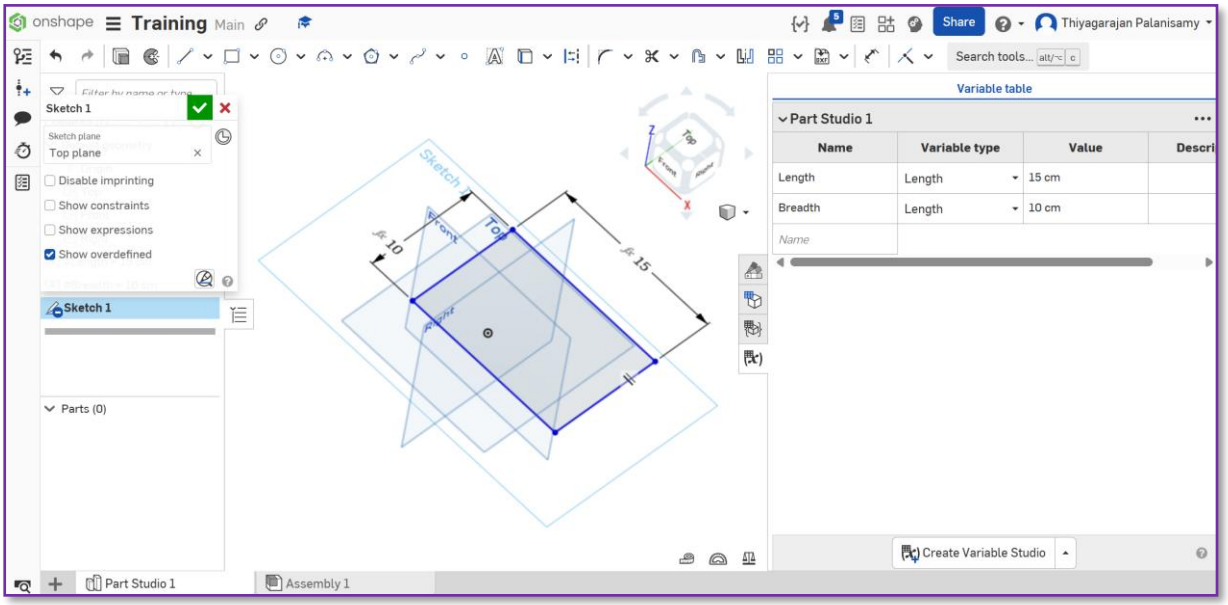
2. Create a "**Variable table**" with two entries, **Length** and **Breadth**, and define the dimensions for these variables.

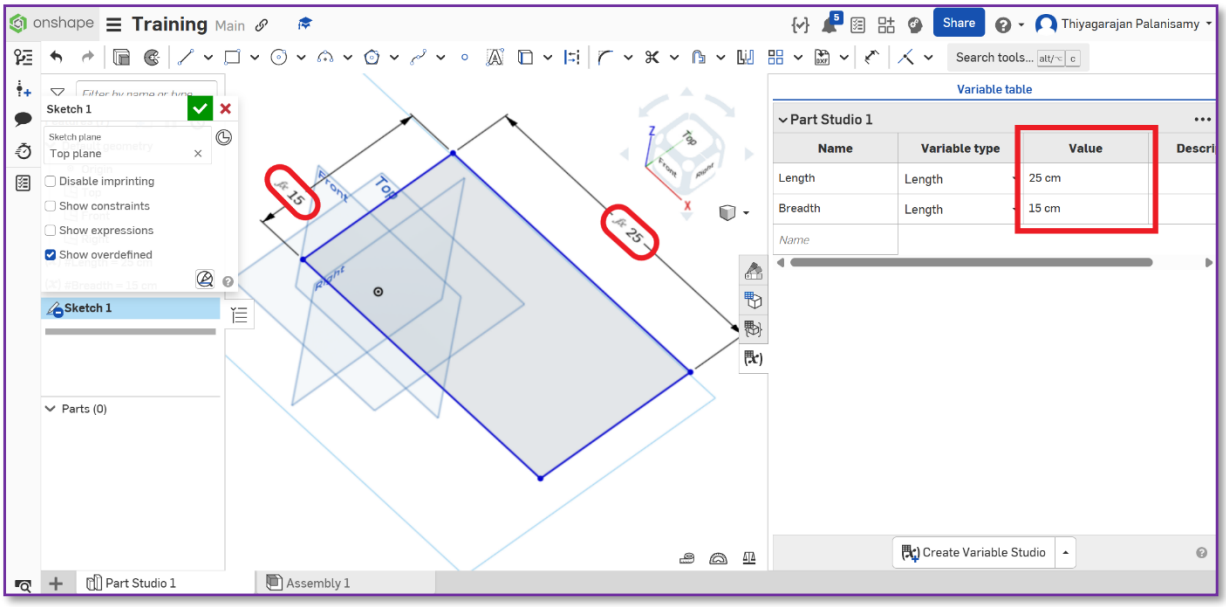


3. Assign the variable table to the sketch dimensions using # **as a prefix**. Double-click on one of the dimensions, type #, and choose a parameter from the variable table.

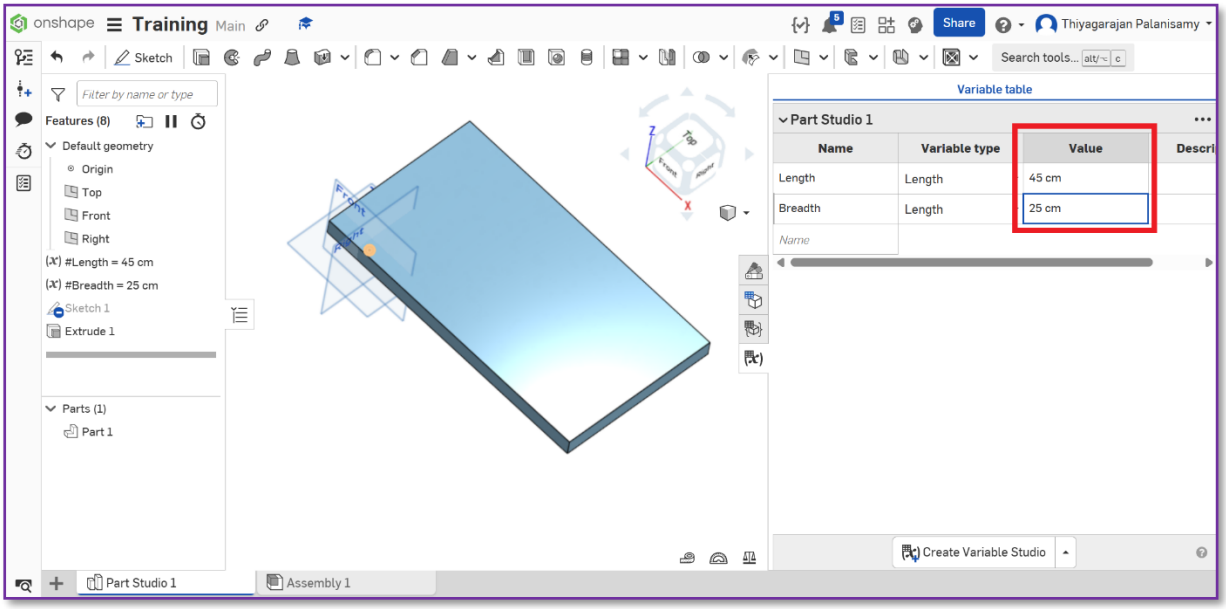


4. Once variables are assigned, a symbol $\boldsymbol{fx}$ will appear next to the dimensions. Now, change the dimension values in the variable table and verify if the sketch updates accordingly



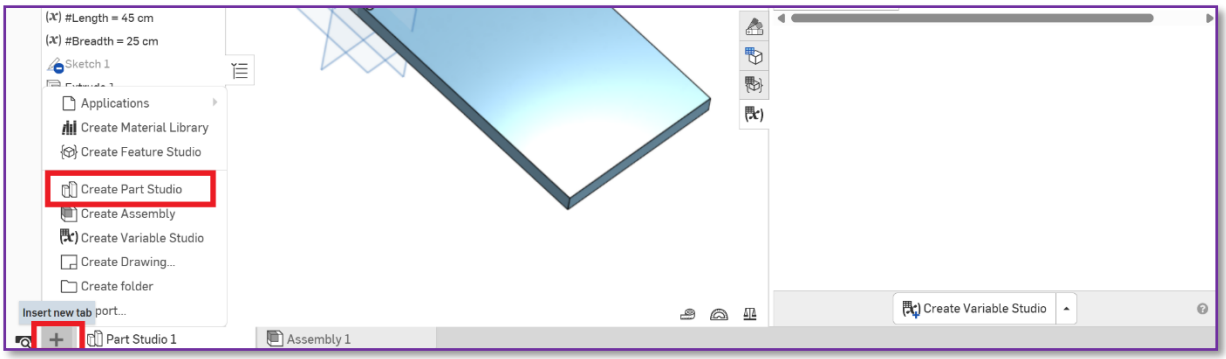


5. The table continues to work even after the sketch is converted into a solid using extrude, revolve, etc.

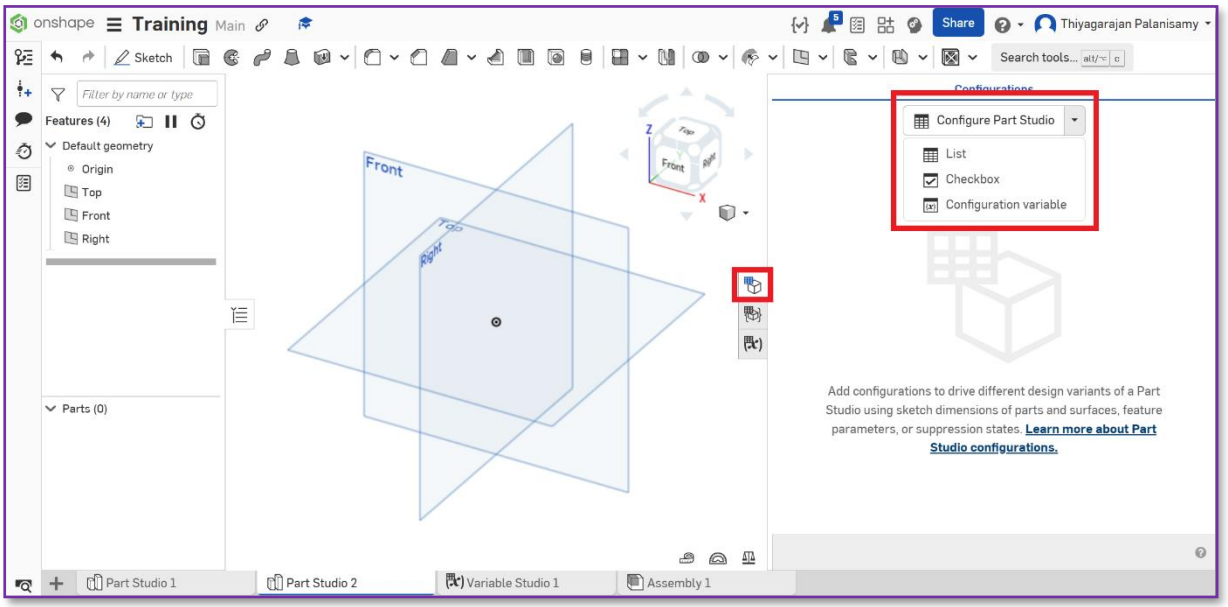


Onshape – Variables

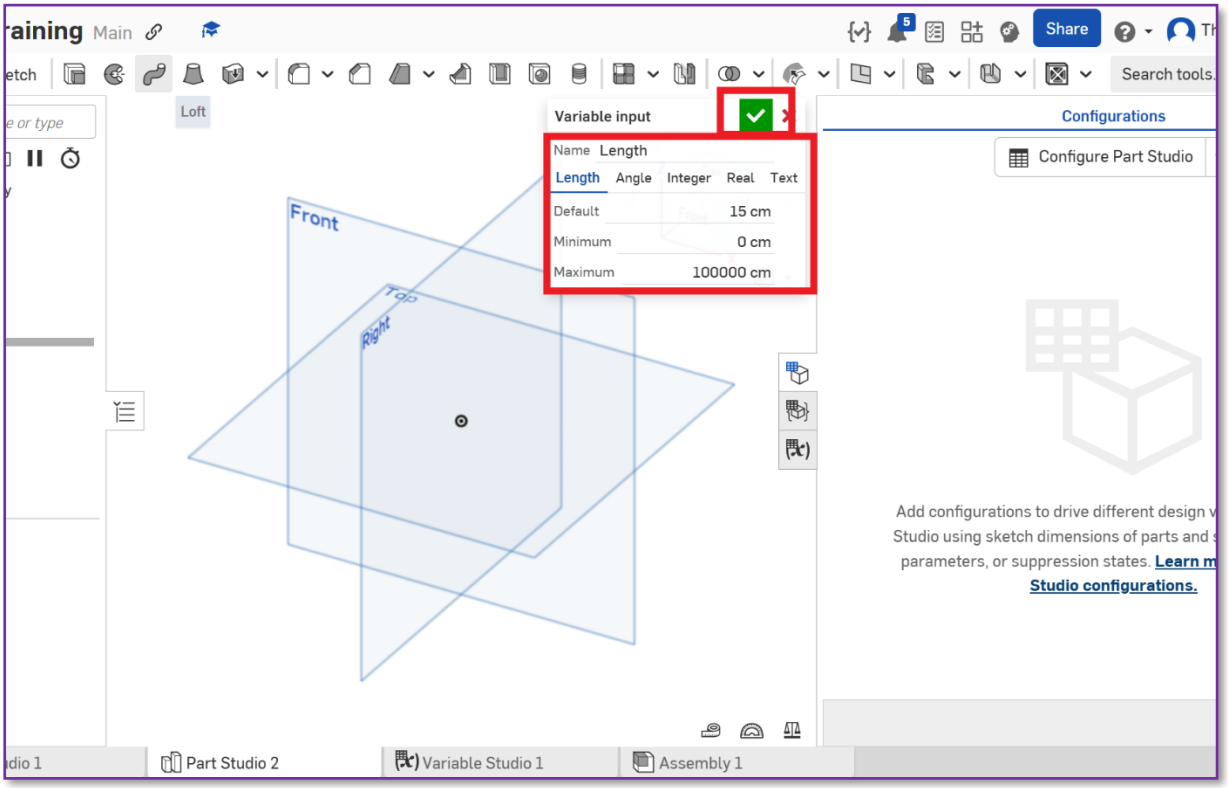
1. Click on "**Insert new tab**" and choose "**Create Part Studio**".



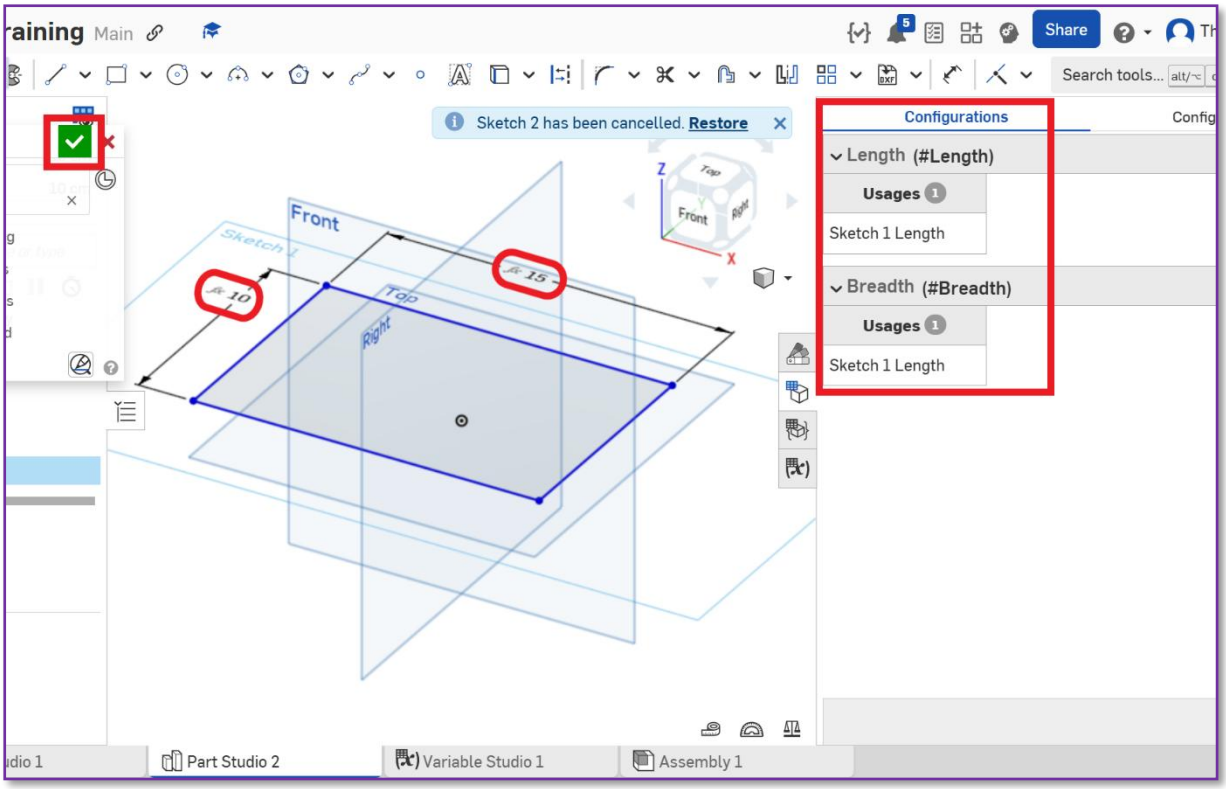
2. Click on “**Configuration panel**” and under **“Configuration Part Studio”** choose “**Configuration variable**”.



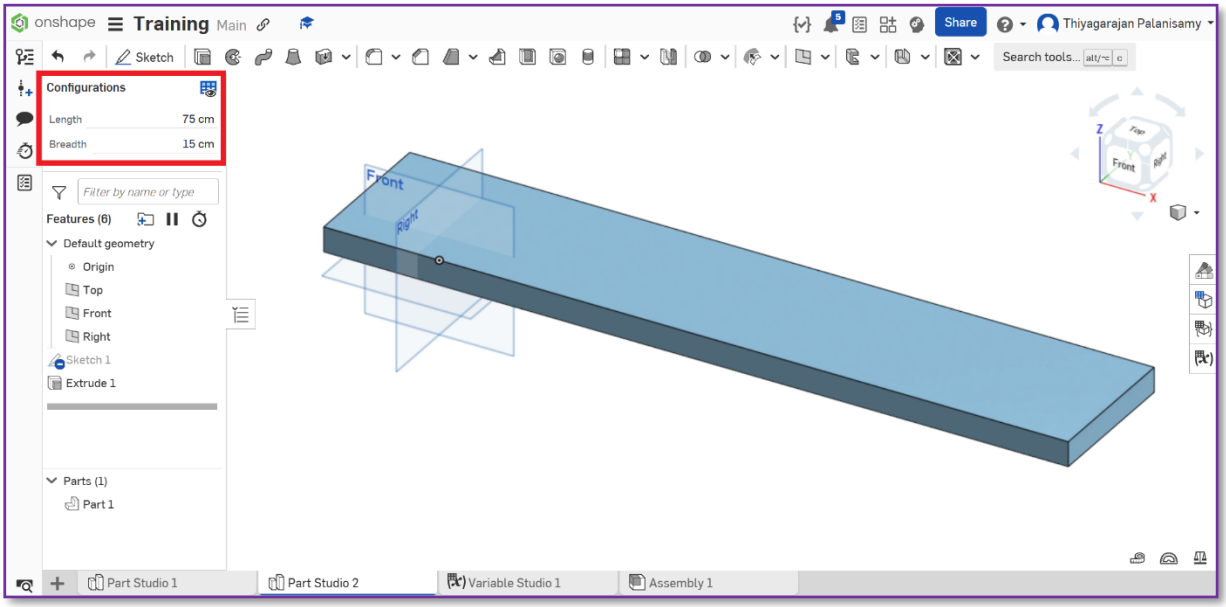
3. Create two variables named Length and Breadth. You can change the default values for both variables as needed.



4. Draw a sketch and assign the variables to the dimensions using the # prefix, as demonstrated in the previous topic in Step 3.

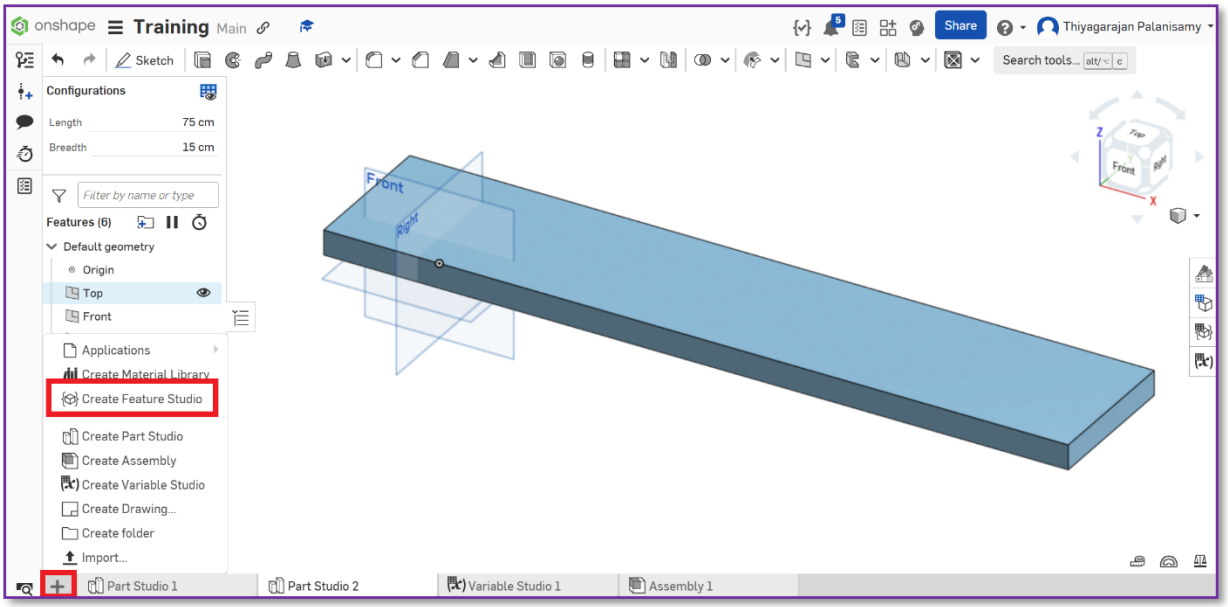


5. Now, the variables can be directly modified using an input form on the left panel, instead of changing them in the variable table.

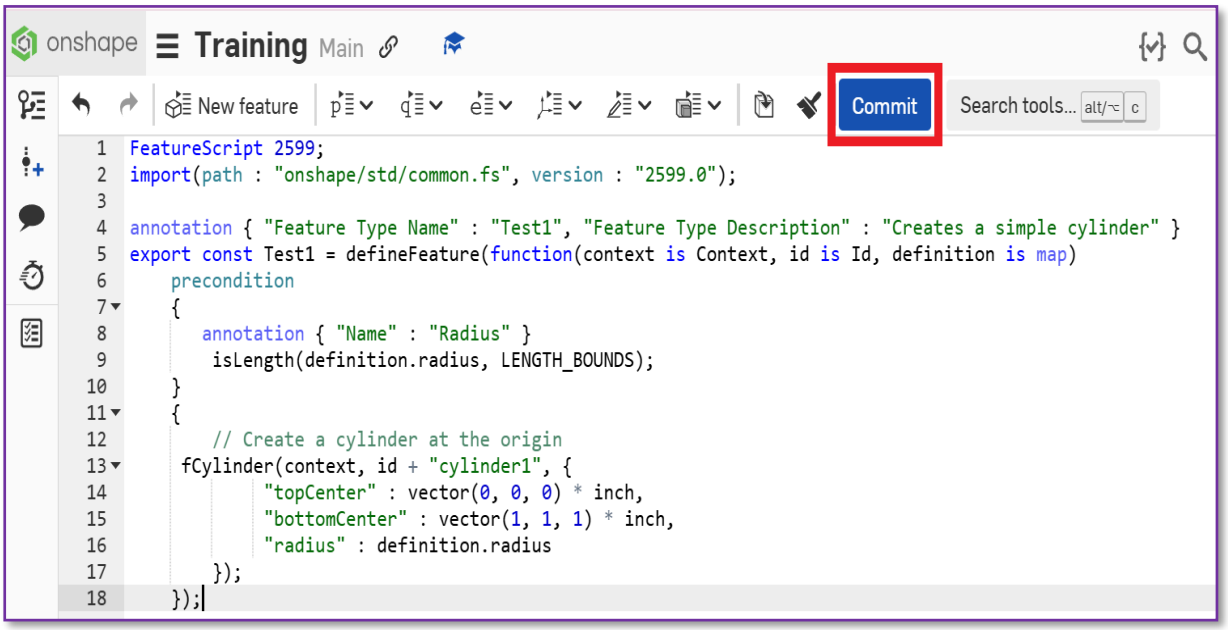


Onshape – Feature Studio

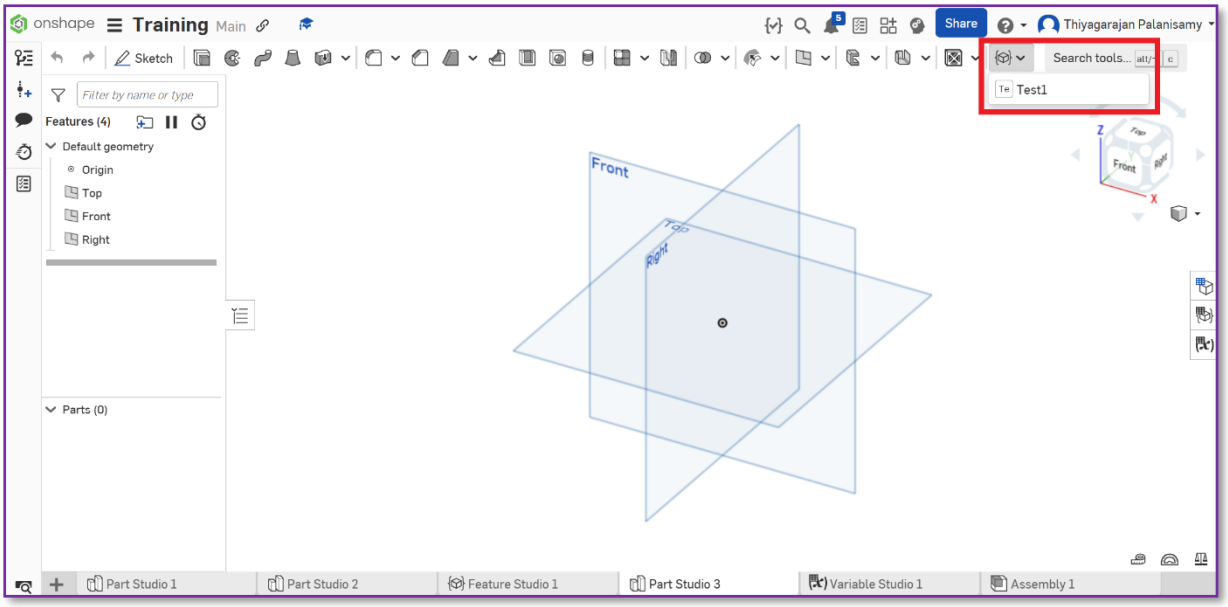
1. Click on "**Insert new tab**" and choose "**Create Feature Studio**".



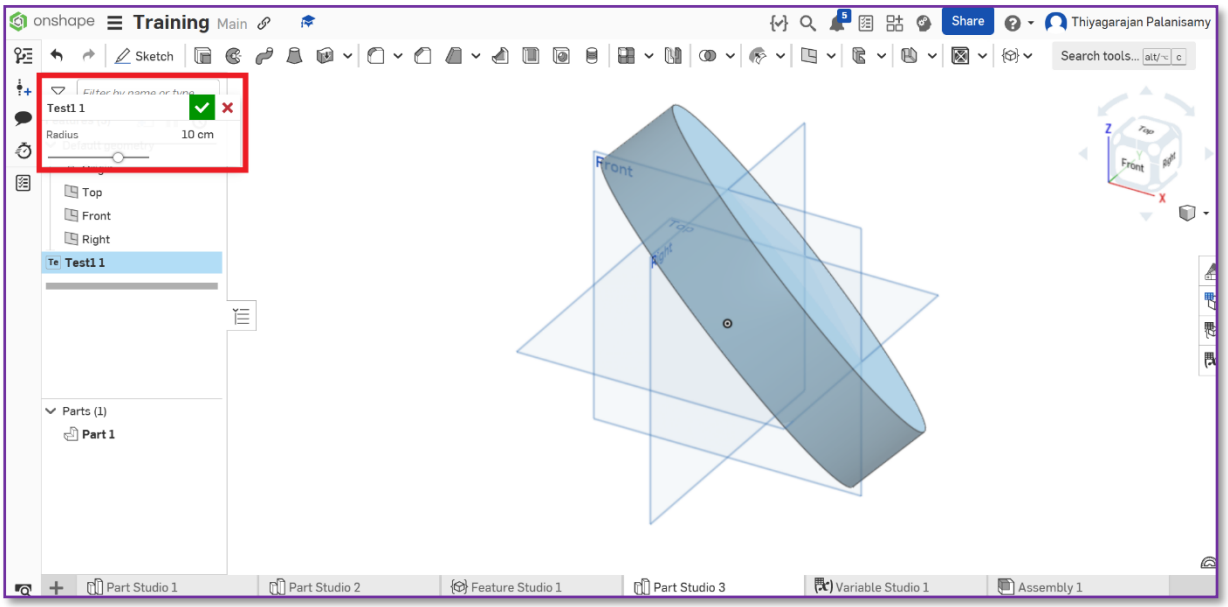
2. Copy the code from the shared document [Onshape Featurescript Samplecode Cylinder.txt] and paste it into the Onshape Feature Studio editor. Click on "**Commit**" to apply the code.

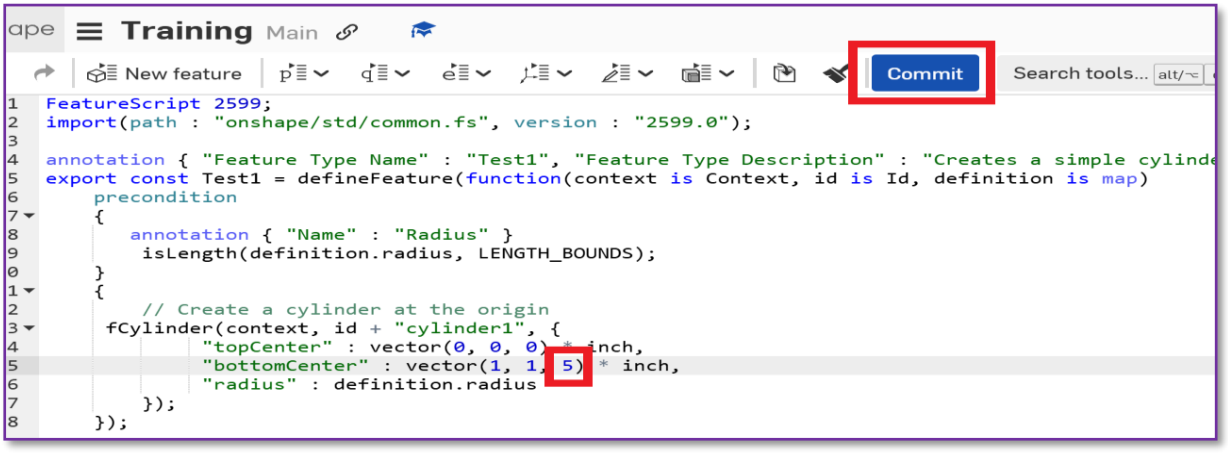


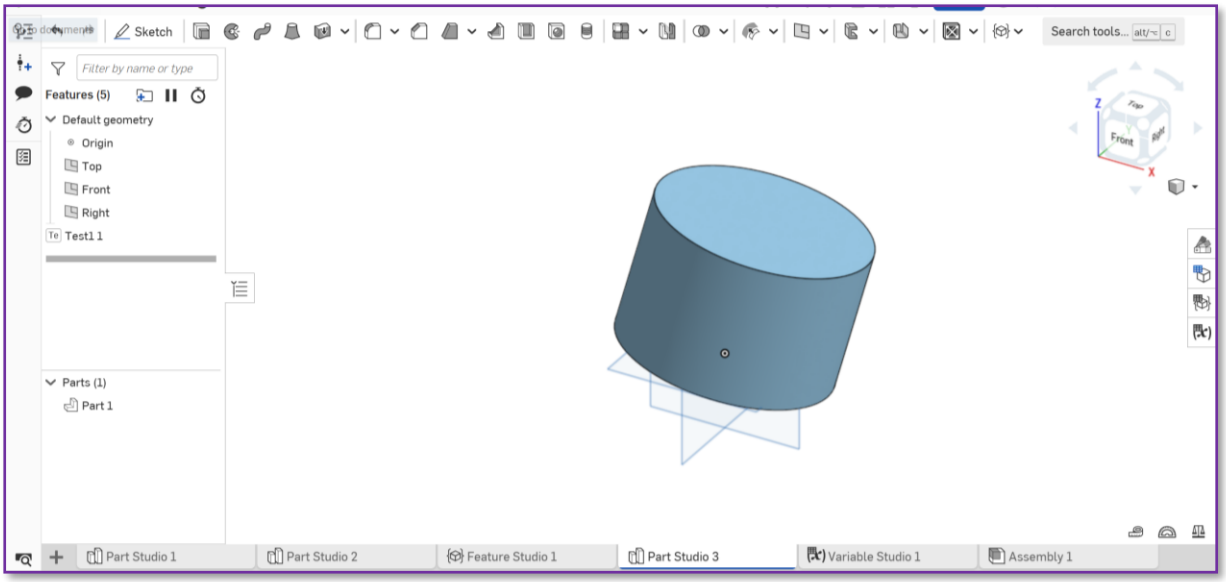
3. Create a **new Part Studio** tab, click on the **"Custom Features"** dropdown, and select **"Test1,"** which corresponds to the code entered in the Feature Studio.



4. A cylinder will be created at the vertex (0,0,0) using the Feature Studio code. The radius of the cylinder can be defined, and its depth can be adjusted by modifying the Z vertex coordinate in the code.





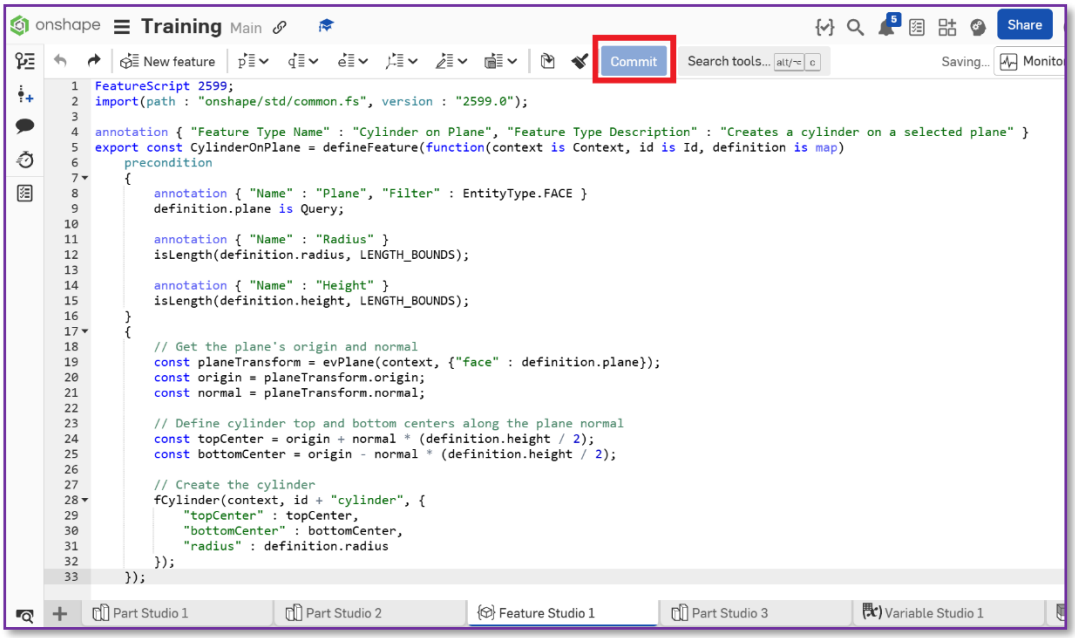


5. Apply the same method to the next two codes available in the shared drive:
   - Onshape_Featurescript_Samplecode_Cuboid.txt
   - Onshape_Featurescript_Samplecode_Cylinderonaplane.txt

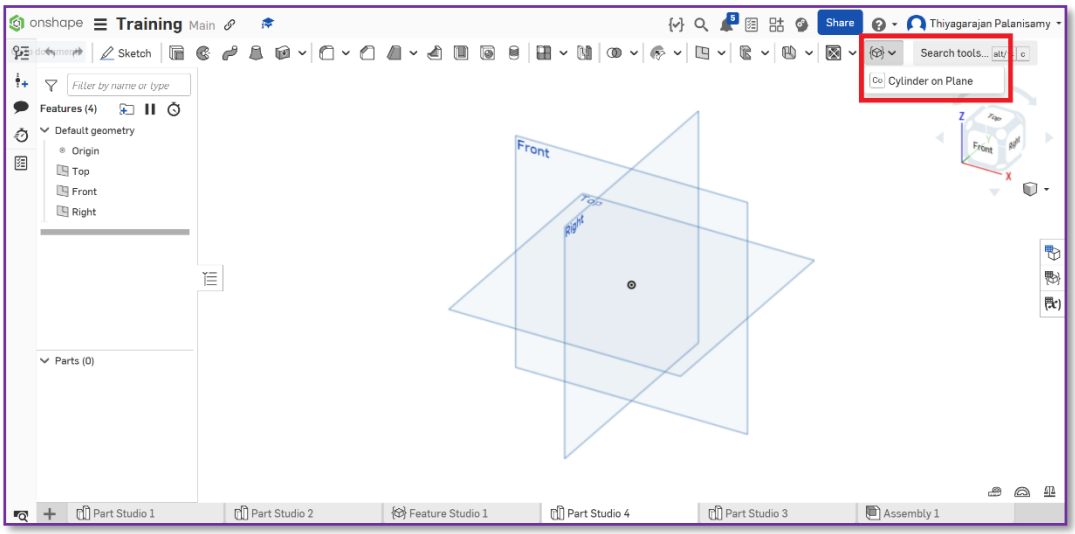
6. The Feature Studio codes can also be generated using ChatGPT. However, keep in mind that the available data is limited, and the codes may require a few iterations of corrections to run without errors.
   Follow these steps to generate Feature Studio code using ChatGPT:
   - Copy the code from Onshape_Featurescript_Samplecode_Cylinder.txt
   - Start a new chat in ChatGPT, paste the code into the prompt, and add "Learn this code" (e.g., {Code} - Learn this code).

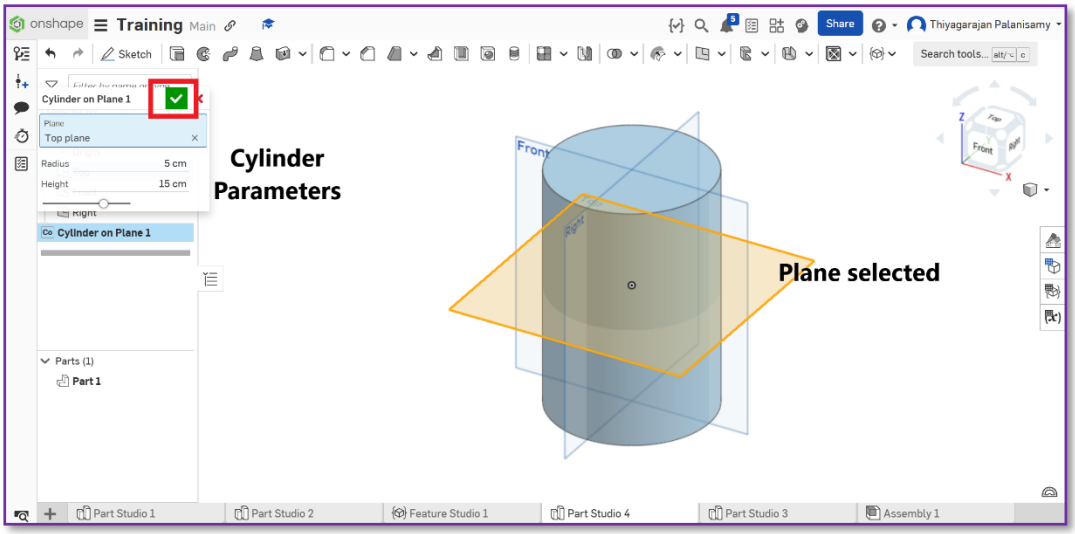
```
FeatureScript 2599;
import(path : "onshape/std/common.fs", version : "2599.0");

annotation { "Feature Type Name" : "Test1", "Feature Type
Description" : "Creates a simple cylinder" }
export const Test1 = defineFeature(function(context is Context, id
is Id, definition is map)
    precondition
    {
        annotation { "Name" : "Radius" }
        isLength(definition.radius, LENGTH_BOUNDS);
    }
    {
        // Create a cylinder at the origin
    fCylinder(context, id + "cylinder1", {
            "topCenter" : vector(0, 0, 0) * inch,
            "bottomCenter" : vector(1, 1, 1) * inch,
            "radius" : definition.radius
        });
    }); - Learn this code
```

- Enter the prompt: "**Modify the Feature Script code to generate the cylinder on a selected plane.**"
- Copy the updated code from ChatGPT, paste it into the Onshape Feature Studio editor, and commit the changes.

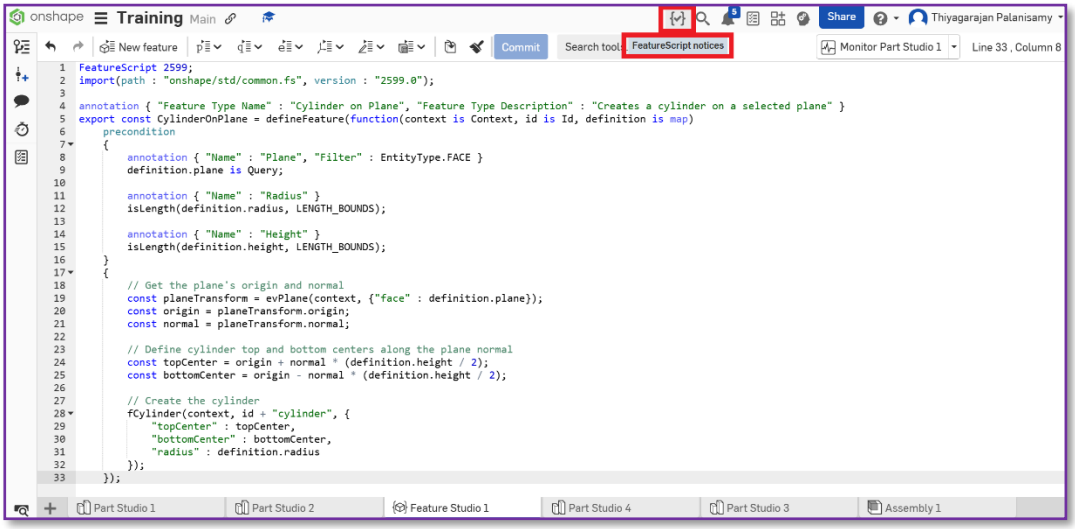


- Create a new Part Studio tab and select **Cylinder on Plane** Feature to verify if the code functions correctly.

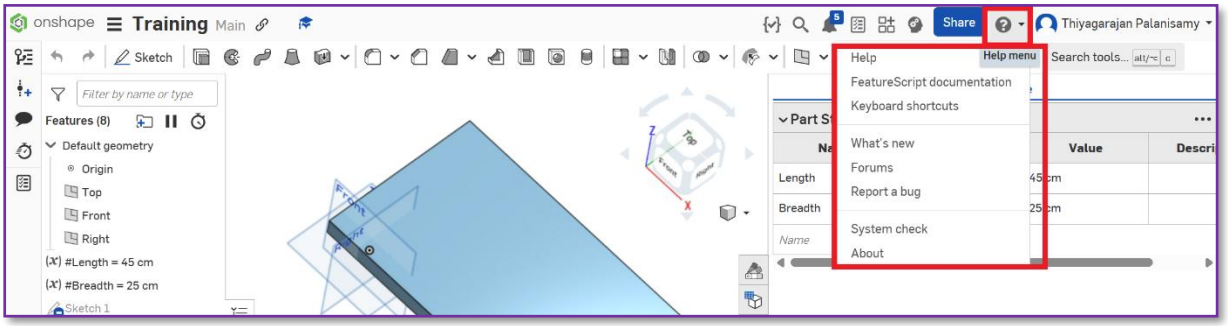




- Use the "**FeatureScript notices**" option for code debugging. It helps identify errors and warnings in the Feature Studio script, making troubleshooting easier.



7. Click on the "**Help**" menu to access **Feature Documentation, Shortcuts, Onshape version**, and other useful resources.



8. Onshape FeatureScript Library containing source codes https://cad.onshape.com/documents/12312312345abcabcabcdeff/w/a855e4161c814f2e9ab3698a/e/daec16bcbf3755b1bc852153